

NASA Technical Memorandum 103800

Viscous Three-Dimensional Calculations of Transonic Fan Performance

Rodrick V. Chima
Lewis Research Center
Cleveland, Ohio

Prepared for the
77th Symposium of the Propulsion and Energetics Panel entitled
CFD Techniques for Propulsion Applications
sponsored by AGARD
San Antonio, Texas, May 27-31, 1991



Viscous Three-Dimensional Calculations of Transonic Fan Performance

Rodrick V. Chima
NASA Lewis Research Center
Cleveland, Ohio, 44135

ABSTRACT

A three-dimensional flow analysis code has been used to compute the design speed operating line of a transonic fan rotor, and the results have been compared with experimental data. The code is an explicit finite difference code with an algebraic turbulence model. It is described in detail in the paper. The transonic fan, designated rotor 67, has been tested experimentally at NASA Lewis Research Center with conventional aerodynamic probes and with laser anemometry and has been included as one of the AGARD test cases for the computation of internal flows. The experimental data are described briefly. Maps of total pressure ratio and adiabatic efficiency versus mass flow have been computed and are compared with the experimental maps, with excellent agreement between the two. Detailed comparisons between calculations and experiment are made at two operating points, one near peak efficiency and the other near stall. Blade-to-blade contour plots are used to show the shock structure. Comparisons of circumferentially integrated flow quantities downstream of the rotor show spanwise distributions of several aerodynamic parameters. Calculated Mach number distributions are compared with laser anemometer data within the blade row and the wake to quantify the accuracy of the calculations. Finally, particle traces are used to illustrate the nature of the secondary flow in this fan.

INTRODUCTION

One goal of computational fluid dynamics (CFD) for turbomachinery is the prediction of component performance, for example pressure ratio and efficiency. Since a small improvement in engine efficiency can amount to huge savings in yearly fuel costs for a fleet of commercial aircraft, turbomachinery designers are extremely interested in tools that give good quantitative predictions of turbomachinery performance. However, most of the CFD results for turbomachinery published more than a few years ago show only qualitative comparisons with experimental Mach contours or quantitative comparisons with surface pressures. This is not to diminish the importance of these results; in the hands of a good turbomachinery designer a shock location or surface pressure distribution can be extremely useful. However, CFD for turbomachinery has evolved to a

point where it can now provide reasonable predictions of overall performance of isolated blade rows.

In more recent literature several researchers have presented more detailed predictions of turbomachinery performance. Davis et al. (ref. 1) predicted loss buckets for two-dimensional transonic compressor cascades. Chevrin and Vuillez (ref. 2) have predicted loss and exit flow angle for turbine and fan cascades using the two-dimensional code of Cambier et al. (ref. 3). Boyle (ref. 4) investigated effects of turbulence modeling on turbine blade heat transfer predictions using the two-dimensional code of Chima (ref. 5). In three dimensions, Pierzga and Wood (ref. 6) predicted the static pressure ratio versus mass flow curve of a transonic fan using Denton's code (ref. 7), and Dawes (ref. 8) has predicted exit total pressure and temperature distributions in a multistage turbine. Adamczyk, et al. (ref. 9) investigated the effects of tip clearance on stall for the fan considered in the present work and predicted pressure ratios and efficiencies for that fan. In reference 10 Chima used the present code to predict the overall efficiency of an annular turbine stator.

There are many reasons for the scarcity of turbomachinery performance calculations in the early literature. One is that pressure field calculations are relatively independent of viscous effects and can be calculated with simple models. Conversely, efficiency and loss calculations are highly dependent on viscous effects and require careful attention to the viscous terms, turbulence modeling, artificial viscosity, and grid resolution for successful calculation. Second, computers capable of performing large viscous flow calculations have become generally available only recently. A final reason is that detailed experimental data are difficult to obtain in turbomachinery because of the small size and high speeds of the components involved. Experimental surface pressures are available from many linear and annular cascade tests, which may account for many of the comparisons appearing in the literature. However, wake surveys and loss data are often available for the same tests, but tend to be overlooked.

With the publication of AGARD Advisory Report No. 275, *Test Cases for Computation of Internal Flows in Aero Engine Components* (ref. 11), researchers now have access to some excellent data for the validation

of CFD codes for turbomachinery.

In the present work the experimental data described by Strazisar et al. (ref. 12) and by Wood et al. (ref. 11, pp. 165–213) was used to validate the three-dimensional Navier-Stokes code first referenced by Chima and Yokota in (ref. 10). The code, RVC3D (Rotor Viscous Code 3D), is described with emphasis on the boundary conditions and artificial viscosity. A new three-dimensional grid code for turbomachinery is also introduced. The test case, a transonic fan rotor (rotor 67, shown in fig. 1) is described briefly.

Several operating points were computed along the 100 percent speed line of the rotor. The computed operating curves of adiabatic efficiency and total pressure ratio versus mass flow are compared with the experimental data. Two operating points are examined in detail: one near peak efficiency and one near stall. At each point qualitative comparisons are made between computed and experimental Mach number contours. Comparisons are then made with laser anemometer measurements within the blade row and across the wake, and with conventional aerodynamic measurements downstream. Finally, particle traces are used to illustrate the nature of the secondary flows in this machine.

GOVERNING EQUATIONS

The Navier-Stokes equations are written in a Cartesian (x, y, z) coordinate system rotating with angular velocity Ω about the x -axis. The rotation introduces source terms in the y and z momentum equations. The Cartesian equations are mapped to a general body-fitted (ξ, η, ζ) coordinate system using standard techniques. A C-type grid was used in the present work, with the ξ -coordinate roughly following the flow, the η -coordinate running blade-to-blade, and the ζ -coordinate running spanwise. The thin-layer approximation is used to drop all viscous derivatives in the ξ -direction. All viscous terms in the cross-channel (η, ζ) plane are retained. The resulting equations are as follows:

$$\partial_t q + J[\partial_\xi \hat{E} + \partial_\eta \hat{F} + \partial_\zeta \hat{G} - \text{Re}^{-1}(\partial_\eta \hat{F}_V + \partial_\zeta \hat{G}_V)] = H \quad (1)$$

where

$$q = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ e \end{bmatrix} \quad \hat{E} = J^{-1} \begin{bmatrix} \rho U' \\ \rho u U' + \xi_x p \\ \rho v U' + \xi_y p \\ \rho w U' + \xi_z p \\ e U' + p U \end{bmatrix}$$

$$H = \Omega \begin{bmatrix} 0 \\ 0 \\ -\rho w \\ \rho v \\ 0 \end{bmatrix} \quad \hat{F} = J^{-1} \begin{bmatrix} \rho V' \\ \rho u V' + \eta_x p \\ \rho v V' + \eta_y p \\ \rho w V' + \eta_z p \\ e V' + p V \end{bmatrix}$$

$$\hat{G} = J^{-1} \begin{bmatrix} \rho W' \\ \rho u W' + \zeta_x p \\ \rho v W' + \zeta_y p \\ \rho w W' + \zeta_z p \\ e W' + p W \end{bmatrix} \quad (2)$$

Again, these equations are written in a rotating Cartesian coordinate system attached to the moving blade. The code solves for the *absolute* velocity components u, v , and w , which point in the x, y , and z coordinate directions, respectively. The *relative* velocity components u', v' , and w' are defined with respect to the same rotating coordinate system by subtracting the appropriate components of the blade speed from the absolute velocities, giving

$$\begin{aligned} u' &= u \\ v' &= v - \Omega z \\ w' &= w + \Omega y \end{aligned} \quad (3)$$

The relative contravariant velocity components are given by

$$\begin{aligned} U' &= \xi_x u + \xi_y v' + \xi_z w' \\ V' &= \eta_x u + \eta_y v' + \eta_z w' \\ W' &= \zeta_x u + \zeta_y v' + \zeta_z w' \end{aligned} \quad (4)$$

Note that although $u' = u$, $U' \neq U$. The energy and static pressure are given by

$$e = \rho \left[C_v T + \frac{1}{2}(u^2 + v^2 + w^2) \right] \quad (5)$$

$$p = (\gamma - 1) \left[e - \frac{1}{2}\rho(u^2 + v^2 + w^2) \right] \quad (6)$$

Using Stokes' hypothesis, $\lambda = -\frac{2}{3}\mu$, the viscous flux \hat{F}_V can be written as follows:

$$\hat{F}_V = J^{-1} \mu [0, F_2, F_3, F_4, F_5]^T \quad (7)$$

where

$$\begin{aligned} F_2 &= C_1 \partial_\eta u + C_2 \eta_x + C_3 \partial_\zeta u - C_4 \eta_x + C_5 \zeta_x \\ F_3 &= C_1 \partial_\eta v + C_2 \eta_y + C_3 \partial_\zeta v - C_4 \eta_y + C_5 \zeta_y \\ F_4 &= C_1 \partial_\eta w + C_2 \eta_z + C_3 \partial_\zeta w - C_4 \eta_z + C_5 \zeta_z \\ F_5 &= \frac{\mu \gamma}{\text{Pr}} (C_1 \partial_\eta (C_v T) + C_3 \partial_\zeta (C_v T)) \\ &\quad + u F_2 + v F_3 + w F_4 \end{aligned} \quad (8)$$

and

$$\begin{aligned} C_1 &= \eta_x^2 + \eta_y^2 + \eta_z^2 \\ C_2 &= \frac{1}{3}(\eta_x \partial_\eta u + \eta_y \partial_\eta v + \eta_z \partial_\eta w) \\ C_3 &= \eta_x \zeta_x + \eta_y \zeta_y + \eta_z \zeta_z \\ C_4 &= \frac{2}{3}(\zeta_x \partial_\zeta u + \zeta_y \partial_\zeta v + \zeta_z \partial_\zeta w) \\ C_5 &= \eta_x \partial_\zeta u + \eta_y \partial_\zeta v + \eta_z \partial_\zeta w \end{aligned} \quad (9)$$

Terms multiplied by C_1 and C_2 lead to nonmixed, second derivative viscous terms like $u_{\eta\eta}$, while terms multiplied by C_3 – C_5 lead to mixed-derivative terms like $u_{\eta\zeta}$. The viscous flux vector \hat{G} can be written similarly by interchanging directions η and ζ and replacing F with G everywhere.

Metric terms are defined using the following relations:

$$J \begin{bmatrix} \xi_x & \eta_x & \zeta_x \\ \xi_y & \eta_y & \zeta_y \\ \xi_z & \eta_z & \zeta_z \end{bmatrix} = \begin{bmatrix} y_\eta z_\zeta - y_\zeta z_\eta & y_\zeta z_\xi - y_\xi z_\zeta & y_\xi z_\eta - y_\eta z_\xi \\ x_\zeta z_\eta - x_\eta z_\zeta & x_\xi z_\zeta - x_\zeta z_\xi & x_\eta z_\xi - x_\xi z_\eta \\ x_\eta y_\zeta - x_\zeta y_\eta & x_\zeta y_\xi - x_\xi y_\zeta & x_\xi y_\eta - x_\eta y_\xi \end{bmatrix} \quad (10)$$

where

$$J = (x_\xi y_\eta z_\zeta + x_\zeta y_\xi z_\eta + x_\eta y_\zeta z_\xi - x_\xi y_\zeta z_\eta - x_\eta y_\xi z_\zeta - x_\zeta y_\eta z_\xi)^{-1} \quad (11)$$

Terms like x_ξ , x_η , etc. are found using second order central differences. The Jacobian is computed using equation (11) and stored for the entire grid. All other metric terms are computed as needed using equation (10).

The equations are nondimensionalized by arbitrary reference quantities (here, the inlet total density ρ_{0ref} and the total sonic velocity c_{0ref} were used), and the Reynolds number Re and Prandtl number Pr are defined in terms of these quantities. The equations assume that the specific heats C_p and C_v and Prandtl number are constant, that Stoke's hypothesis is valid, and that the effective viscosity for turbulent flows may be written as

$$\mu_{eff} = \mu_{lam} + \mu_{turb} \quad (12)$$

where the laminar viscosity is calculated using a power law function of temperature:

$$\frac{\mu_{lam}}{\mu_{ref}} = \left(\frac{T}{T_{ref}} \right)^n \quad (13)$$

with $n = 2/3$ for air. The turbulent viscosity μ_{turb} is computed using an adaptation of the Baldwin-Lomax turbulence model (ref. 13) on cross-channel planes. Briefly, the model is applied independently in the blade-to-blade and spanwise directions, and the resulting turbulent viscosities are added vectorally. The two-dimensional Buleev length scale based on the distance from the hub and blade is used. The vorticity is calculated in the absolute frame, but the wall shear and wake velocities are calculated in the relative frame. See reference 10 for details on the three-dimensional implementation.

BOUNDARY CONDITIONS

Many boundary conditions for turbomachinery are best expressed in cylindrical coordinates, but the

code is formulated in Cartesian coordinates. For the boundary conditions, coordinate systems are transformed as needed using:

$$\begin{aligned} v &= (v_r y + v_\theta z)/r \\ w &= (v_r z - v_\theta y)/r \\ v_r &= (vy + wz)/r \\ v_\theta &= (vz - wy)/r \\ r &= \sqrt{y^2 + z^2} \end{aligned} \quad (14)$$

At the inflow boundary the total temperature, total pressure, whirl, and meridional flow angle were specified, and the upstream-running Riemann invariant was extrapolated from the interior. The inlet total temperature T_0 was specified as a constant (standard conditions). The inlet total pressure was specified as a constant in the core flow (also standard conditions) and reduced in the endwall regions according to a $1/7$ power law velocity profile, with the inlet boundary layer heights estimated from the experimental data to be 12 mm on both the hub and the casing. At the inlet v_θ was set to zero, and v_r was chosen to make the flow tangent to the meridional projection of the inlet grid lines. The upstream-running Riemann invariant R^- based on the total absolute velocity $Q = \sqrt{u^2 + v^2 + w^2}$ was calculated at the first interior point and extrapolated to the inlet. The Riemann invariant is given by

$$R^- = Q - \frac{2c}{\gamma - 1} \quad (15)$$

The total velocity is found from T_0 and R^- using

$$Q = \frac{(\gamma - 1)R^- + \sqrt{2(1 - \gamma)(R^-)^2 + 4(\gamma + 1)C_p T_0}}{\gamma + 1} \quad (16)$$

The velocity components are then decoupled algebraically, and the density is found from P_0 , T_0 , and Q using an isentropic relation.

At the exit the hub static pressure is specified, and ρ , ρu , ρv , and ρw are extrapolated from the interior. The local static pressure is found by integrating the axisymmetric radial equilibrium equation:

$$\frac{dp}{dr} = \frac{\rho v_\theta^2}{r} = \frac{\rho}{r^3} (vz - wy)^2 \quad (17)$$

A periodic C-grid was used in the present work. The periodic boundary was solved by setting periodic flow conditions (in terms of cylindrical velocity components) on a dummy grid line outside the boundary. Fourth-difference artificial dissipation terms are neglected on the outer boundary so that only one dummy grid line is needed.

On the blade surface and the rotating part of the hub, $U' = V' = W' = 0$. The hub was specified to be stationary 13.4 mm upstream of the leading edge and 3.35 mm downstream of the trailing edge by setting

$u = v = w = 0$. The tip casing was assumed to be stationary, that is, the blade was assumed to scrape along the casing with zero tip clearance. Surface pressures were found from the normal momentum equation. On the hub ($\zeta = 1$) and tip ($\zeta = \zeta_{max}$)

$$(\zeta_x \xi_x + \zeta_y \xi_y + \zeta_z \xi_z) \partial_\xi p + (\zeta_x \eta_x + \zeta_y \eta_y + \zeta_z \eta_z) \partial_\eta p + (\zeta_x^2 + \zeta_y^2 + \zeta_z^2) \partial_\zeta p = -\rho[\Omega(\zeta_y w - \zeta_z v)] \quad (18)$$

On the blades ($\eta = 1$) the normal momentum equation can be found by replacing ζ with η everywhere in equation (18).

Surface temperatures were found from an adiabatic wall condition implemented as in equation (18) with p replaced by T and a right-hand side of zero.

MULTISTAGE RUNGE-KUTTA SCHEME

The governing equations are discretized using a node-centered finite difference scheme. Second-order central differences are used throughout. The multistage Runge-Kutta scheme developed by Jameson, Schmidt, and Turkel (ref. 14) is used to advance the flow equations in time from an initial guess to a steady state. If equation (1) is rewritten as

$$\partial_t q = -J[R_I - (R_V + D)] \quad (19)$$

where R_I is the inviscid residual including the source term, R_V is the viscous residual, and D is an artificial dissipation term described in the next section, then the multistage Runge-Kutta algorithm can be written as follows:

$$\begin{aligned} q_0 &= q_n \\ q_1 &= q_0 - \alpha_1 J \Delta t [R_I q_0 - (R_V + D) q_0] \\ &\vdots \\ q_k &= q_0 - R_k \\ &= q_0 - \alpha_k J \Delta t [R_I q_{k-1} - (R_V + D) q_0] \\ q_{n+1} &= q_k \end{aligned} \quad (20)$$

Here a standard four-stage scheme was used, with $\alpha_i = 1/4, 1/3, 1/2, 1$. For efficiency both the physical and artificial dissipation terms are calculated only at the first stage, then are held constant for subsequent stages.

ARTIFICIAL DISSIPATION

The dissipative term D in equation (19) is similar to that used by Jameson et al. (ref. 14). It is given by

$$Dq = (D_\xi + D_\eta + D_\zeta) q \quad (21)$$

where the ξ -direction operator is given by

$$D_\xi q = C_\xi (V_2 q_{\xi\xi} - V_4 q_{\xi\xi\xi\xi}) \quad (22)$$

The terms V_2 and V_4 are given by

$$\begin{aligned} V_2 &= \mu_1 + \mu_2 \max(\nu_{i+1}, \nu_i, \nu_{i-1}) \\ V_4 &= \max(0, \mu_4 - V_2) \end{aligned} \quad (23)$$

where

$$\nu_i = \frac{|P_{i+1} - 2P_i + P_{i-1}|}{\min(P_{in}, |P_{i+1} + 2P_i + P_{i-1}|)} \quad (24)$$

and subscript i corresponds to the ξ direction, etc.

The constant μ_1 scales a first-order artificial viscosity that is useful for maintaining stability at startup. The constant μ_2 scales a first-order artificial artificial viscosity that is switched on at shocks detected by equation (24). The denominator in equation (24) is normally constant at the inlet pressure P_{in} , making the operator roughly symmetric across shocks. The more common term $|P_{i+1} + 2P_i + P_{i-1}|$ is included to switch on the second-difference dissipation when the pressure becomes very small, usually due to numerical problems. The constant μ_4 scales a uniform third-order artificial viscosity that is switched off at shocks by equation (23). In this work μ_1 was set to $1/4$ for the first 200 iterations and zero thereafter, $\mu_2 = 1/2$, and $\mu_4 = 1/32$.

C is an arbitrary coefficient that can have a large impact on the stability and accuracy of the solution. The subscript indicates that C may be different in each direction.

In reference 10 a directionally homogeneous coefficient was used for three-dimensional problems. To minimize dissipation in viscous regions C was reduced to zero linearly over several grid points near walls. In reference 5 a directionally biased coefficient was used for two-dimensional problems. The dissipation was proportional to the grid spacing in each direction, thus reducing it across finely gridded viscous regions. This worked well in two dimensions but did not generalize to three dimensions.

Martinelli and Jameson (ref. 15) proposed a directionally biased coefficient that works well but can still have large directional variations on highly stretched three-dimensional grids. Kunz and Lakshminarayana (ref. 16) proposed a modified form of the Martinelli and Jameson coefficient:

$$\begin{aligned} C_\xi &= \frac{1}{J \Delta t_\xi} \left(1 + \frac{\Delta t_\xi}{\Delta t_\eta} + \frac{\Delta t_\xi}{\Delta t_\zeta} \right)^\sigma \\ C_\eta &= \frac{1}{J \Delta t_\eta} \left(1 + \frac{\Delta t_\eta}{\Delta t_\xi} + \frac{\Delta t_\eta}{\Delta t_\zeta} \right)^\sigma \end{aligned} \quad (25)$$

etc., where $\sigma = 2/3$.

In the present work the one-dimensional time steps are written as follows:

$$\Delta t_\xi = \left(V_\xi \sqrt{\xi_x^2 + \xi_y^2 + \xi_z^2} \right)^{-1} \quad (26)$$

etc., where the normalized velocity scale V_ξ is approximated as one. Dissipation coefficients given by equations (25) and (26) seem to give a good distribution of the dissipative terms in each direction and have been used throughout this work.

THREE-DIMENSIONAL STABILITY LIMIT

The following expression is used for the time step:

$$\Delta t \leq \frac{\lambda^*}{l_x|u| + l_y|v'| + l_z|w'| + \sqrt{c^2(l_x^2 + l_y^2 + l_z^2) + \Omega^2}}$$

where

$$\begin{aligned} l_x &= |\xi_x| + |\eta_x| + |\zeta_x| \\ l_y &= |\xi_y| + |\eta_y| + |\zeta_y| \\ l_z &= |\xi_z| + |\eta_z| + |\zeta_z| \end{aligned} \quad (27)$$

and λ^* is the maximum Courant number for the particular multistage scheme. For the standard four-stage scheme $\lambda^* \approx 2.8$.

To accelerate convergence to a steady state, the maximum permissible time step at each grid point was used giving a constant Courant number everywhere. The time step was updated every 50 iterations.

IMPLICIT RESIDUAL SMOOTHING

To further accelerate convergence it is desirable to use a time step even larger than the stability limit given by equation (27). To maintain stability, the residual calculated in equation (19) is smoothed after each Runge-Kutta stage by an implicit smoothing operator, that is,

$$(1 - \epsilon_\xi \delta_{\xi\xi})(1 - \epsilon_\eta \delta_{\eta\eta})(1 - \epsilon_\zeta \delta_{\zeta\zeta})\bar{R}_k = R_k \quad (28)$$

where $\delta_{\xi\xi}$, $\delta_{\eta\eta}$, and $\delta_{\zeta\zeta}$ are standard second difference operators and ϵ_ξ , ϵ_η , and ϵ_ζ are smoothing parameters.

Linear stability analysis shows that the Runge-Kutta scheme may be made unconditionally stable using implicit residual smoothing if the smoothing parameters ϵ are made sufficiently large (ref. 17). In one dimension

$$\epsilon \geq \frac{1}{4} \left[\left(\frac{\lambda}{\lambda^*} \right)^2 - 1 \right] \quad (29)$$

gives unconditional stability if λ^* is the Courant limit of the unsmoothed scheme, and λ is a larger operating Courant number. In three dimensions different ϵ 's may be used in each direction, and their magnitudes may often be reduced below the value given by equation (29). Courant numbers, $\lambda^* \approx 2.8$ and $\lambda = 5.0$, and smoothing parameters, $\epsilon_\xi = 0.4$, $\epsilon_\eta = 0.55$, and $\epsilon_\zeta = 0.45$, were used in the present work.

COMPUTATIONAL GRID

A three-dimensional grid code for turbomachinery has been developed by the author (currently unpublished.) The code, called TCGRID for Turbomachinery C-GRID, generates three-dimensional C-type grids using the following technique:

1. A coarse, equally spaced meridional grid is generated between the specified hub and tip.
2. Blade coordinates are found at the meridional grid points by interpolation of the input blade geometry.

3. Two-dimensional blade-to-blade grids are generated along the meridional grid lines in $(m, \bar{r}\theta)$ coordinates using a version of the GRAPE code developed by Steger and Sorenson (ref. 18). Here, m is the arc length along the meridional surface, and \bar{r} is some mean radius. The GRAPE code allows arbitrary specification of inner and outer boundary points, then generates interior points as the solution of a Poisson equation. Forcing terms in the Poisson equation are chosen to maintain the desired grid spacing and angles at the boundaries.

4. The $(m, \bar{r}\theta)$ coordinates are transformed back to (z, r, θ) .

5. The two-dimensional grids are reclustered spanwise using a hyperbolic tangent stretching function to make a full three-dimensional grid.

6. Finally, the (z, r, θ) coordinates are transformed to (x, y, z) and stored in a standard format.

Figure 2 shows the 185x40x49 grid used for rotor 67. The C-shaped grids used here give good resolution of the round leading edge of the blade, as shown in figure 3. The initial grid spacing is about 0.015 mm at the blade, 0.03 mm at the hub and 0.045 mm at the tip.

COMPUTATIONAL DETAILS

All computations were run on the Cray Y-MP computer at NASA Ames Research Center, under support from the Numerical Aerodynamic Simulation (NAS) Project Office.

The grid code required about 3 million words (Mw) of in-core storage and ran in about 15 seconds for the grid shown in figures 2 and 3 (362 600 points.) The flow solver required about 6.5 Mw of storage and 3.6 Mw of solid state device storage (SSD.) The SSD storage is used to hold the old solution q_0 and the dissipative terms $R_V + D$ during the four stages of the multistage scheme (eq. (20)).

The spanwise inlet profile was used as an initial guess, and 250 iterations were run with an exit pressure corresponding to the peak efficiency point. About 20 minutes of CPU time were required for this startup solution. All subsequent calculations were restarted from this solution and run an additional 1550 iterations, requiring about 2.5 CPU hours per case.

EXPERIMENTAL DETAILS

Experimental details are described briefly below. Full details may be found in references 11 and 12.

Test rotor. The test rotor, NASA designation rotor 67, is shown in figure 1. It is the first stage rotor of a two stage fan, with a design pressure ratio of 1.63 at a mass flow of 33.25 kg/sec. The rotor has 22 blades. The tip radius varies from 25.7 cm at the leading edge to 24.25 cm at the trailing edge, and the hub to tip radius ratio varies from 0.375 to 0.478. At the design rotational speed of 16 043 rpm the tip speed is 429 m/sec, and the tip relative Mach number is 1.38.

Aerodynamic performance measurements. The rotor mass flow was determined using a calibrated orifice. Radial surveys of total pressure and temperature, static pressure, and flow angle were made 2.54 cm upstream of the leading edge and 2.02 cm downstream of the trailing edge using conventional probes.

Laser anemometry measurements. A single channel fringe anemometer was used for the measurements described below. Optical access to the fan was through a glass window in the casing. Fluorescent oil droplets with a diameter of about 1.0–1.4 μm were used to seed the flow.

Measurements were made by holding the probe volume fixed in space as the blades rotated by. Measurements were taken in 50 intervals going from the suction surface of one blade to the suction surface of the next blade. In references 11 and 12, and in the present work the term *windows* is used interchangeably with the term *intervals* in referring to laser measurement locations. Within the blade row the blade itself obscures the last few windows. Measurements were taken in 17 consecutive blade passages, then averaged. Only axial and tangential velocities were measured; however, Pierzga and Wood (ref. 6) have shown numerically that neglecting the radial component has a minimal effect on calculated Mach numbers.

RESULTS

The computed total pressure ratio and adiabatic efficiency at 100 percent speed are plotted against normalized mass flow and compared with experimental data in figure 4. The computed and experimental mass flows are normalized by their respective choking mass flows, as suggested by Pierzga and Wood. This normalization removes any uncertainties in the experimental mass flows. The choking mass flow was measured as 34.96 kg/sec using a calibrated orifice, and the computed value was 34.54 kg/sec, a difference of 1.2 percent. The computed pressure ratios and efficiencies agree very well with the experimental data, except that they are slightly high near stall.

Figures 5 to 8 show detailed results at an operating point near peak efficiency. Figure 5 shows a comparison of relative Mach number contours at 10, 30, and 70 percent span from the tip. The experimental contours were drawn from laser anemometry data using smoothing and interpolation procedures outlined in reference 6. At 10 percent span the inlet Mach number is about 1.35, and a bow wave stands ahead of the blade. A weak oblique shock crosses the passage inside the blade row, and a strong normal shock sits near the trailing edge. The flow exits at a Mach number of about 0.95. The flow is qualitatively similar at 30 percent span, except that the shock sits more forward. At 70 percent span the inlet Mach number is about 0.95. A small supersonic bubble forms on the forward portion of the suction surface. It is not clear if this bubble is terminated by a shock.

Figure 6 shows a comparison of computed and measured blade-to-blade profiles of relative Mach number versus laser window number at two axial locations for each of three spanwise locations. The first axial location is within the blade row and the second is within the wake. Again, 50 laser windows are defined from the suction surface of one blade (window 0) to the suction surface of the next blade (window 50.)

Within the blade row the computed profiles agree reasonably well with the laser data and predict the shock locations accurately. At 70 percent span near the pressure surface (laser windows greater than 40), few seed particles were observed, and the data are statistically uncertain.

Within the wake the computed Mach numbers are somewhat low near the tip and get progressively better towards the hub. The computed wake profiles are deeper than the measured profiles. It is thought that this is because the seed particles used for the laser anemometer measurements could not follow the high shear rates found at the center and edges of the wake.

The computational results were averaged blade-to-blade using an "energy average" procedure developed by D. L. Tweedt at NASA Lewis Research Center (unpublished.) On each blade-to-blade grid line the procedure integrates the mass flow, radial and tangential momentum, total enthalpy, and ideal total enthalpy $P_0^{\frac{\gamma-1}{\gamma}}$. The radial distributions were mass averaged spanwise to produce the overall averages shown previously in figure 4. Since the integrations give total conditions directly, they can be expected to give good predictions of efficiency. Other quantities like average static pressure or flow angle are found as nonlinear algebraic combinations of the primitive integrated quantities, and thus may not agree as well with experimental data.

Experimentally, the exit total conditions and flow angle were measured with a self-nulling combination probe, and static pressure was measured using a separate self-nulling wedge probe. Total conditions are thought to be fairly accurate, but flow angle and static pressure accuracy depend on the frequency response and blockage of the instrumentation.

Radial surveys of several aerodynamic parameters measured 2.02 cm downstream of the rotor (at the grid exit) are shown in figure 7. The computed exit total temperatures and total pressures agree very well with the measurements along the span, but the computed static pressures are somewhat high, and computed exit flow angles are two to three degrees high over much of the span.

Figure 8 shows several particle traces to illustrate some of the secondary flows in this fan. The overall view shows particles introduced upstream so as to pass over the suction surface of the blade. Most of the particles pass straight through the blade row, but those in the endwall boundary layer roll up into a vortex that

climbs the leading edge. The enlargement of the leading edge shows that the flow in the hub boundary layer sees a high blade incidence and separates near the leading edge. The low-momentum fluid in this separated region is centrifuged radially outwards. Eventually the incidence decreases, and the flow turns abruptly in the streamwise direction. The trailing edge enlargement shows a large separation bubble fed by fluid from the endwall that migrates radially outward and ends up in the wake. The complicated flow pattern in this region may explain the scarcity of laser seed particles observed in this region.

Figures 9 to 12 show detailed results at an operating point near stall. Figure 9 shows a comparison of relative Mach number contours, again at 10, 30, and 70 percent span from the tip. At 10 percent span the inlet Mach number is about 1.4. A normal shock stands ahead of the blade and crosses the passage. The computed shock is somewhat stronger than, and ahead of, the measured shock. The exit Mach number is about 0.85. At 30 percent span the results are similar. The flow at 70 percent span is similar to the peak efficiency case, except that the supersonic bubble is smaller and has a better defined (though smeared) terminating shock.

Figure 10 shows a comparison of relative Mach number profiles, again at two axial locations and three spanwise locations. At the tip the computed shocks are ahead of and stronger than the measured shocks, which accounts for the high predicted pressure ratios near stall. Adamczyk et al. (ref. 9) have shown numerically that interaction of the tip leakage vortex and the tip shock has a large effect on the near-stall performance of this rotor, and that lack of a tip clearance model can account for the discrepancies in shock position and strength seen here. At 70 percent the computed Mach numbers are slightly low, but agree qualitatively with the data.

In the wake, the computed Mach numbers are somewhat low near the tip and get progressively better towards the hub. Again, the computed wakes are deeper than the measured wakes. There is considerable uncertainty in the laser data in the center of the wake at 70 percent span, where computed particle traces shown later indicate a large separation.

Figure 11 compares radial surveys downstream of the rotor. The computed total temperatures and pressures are slightly high along the span, consistent with the high adiabatic efficiency shown in figure 4. Again, the computed static pressures are high. Computed exit flow angles agree well near the tip but are a few degrees high at lower radii.

Figure 12 shows particle traces at the near-stall operating point. The overall view shows more radial migration of the endwall flow than at peak efficiency. At this lower mass flow the blade sees a higher relative incidence than at peak efficiency, causing the flow near the hub to migrate tangentially away from the suc-

tion surface, as shown in the leading edge enlargement. The trailing edge enlargement shows a separation bubble similar to that seen near peak efficiency. The trailing edge separation appears to be slightly larger at the near-stall point.

SUMMARY

A flow analysis code has been developed for three-dimensional viscous flows in turbomachinery. The analysis solves the Navier-Stokes equations written in a general body-fitted coordinate system, including rotation about the x -axis. The thin-layer approximation is made in the streamwise direction but all viscous terms are included in the cross planes. The Baldwin-Lomax eddy-viscosity model is used for turbulent flows.

An explicit multistage Runge-Kutta scheme is used to solve the finite-difference form of the flow equations. A variable time step and implicit residual smoothing are used to accelerate the convergence of the scheme. The code is highly vectorized for the Cray Y-MP, and solutions can be computed on fairly fine grids in 2 to 3 hours.

The code was used to compute the operating map of a transonic fan at design speed and showed good agreement with measured values of total pressure ratio and adiabatic efficiency. The computed results are slightly optimistic near stall, probably due to the lack of a tip clearance model.

Detailed comparisons were made with experimental data at two operating points, one near peak efficiency and one near stall. Comparisons were made with aerodynamic surveys downstream of the fan. In general, exit total temperature and total pressure were predicted quite accurately, but static pressure and flow angle showed some disagreement with the data. Comparisons were also made with laser anemometry data in the blade row and in the wake. Shock location and strength were predicted closely near peak efficiency, but were overpredicted near stall. Predicted wake profiles had about the right location and spread but were much deeper than measured wakes, probably due to lack of resolution in the laser data.

Particle traces showed separated flow at both the leading and trailing edges at both operating points. The leading edge separation is too small to be seen in the laser data. The trailing edge separation may have been suggested indirectly by the scarcity of seed particles in this region. The ability of the code to predict these separated flow features suggests that the code could be used to guide experimental work aimed at resolving these features or to eliminate such features during design.

Overall the code showed very good agreement with a variety of experimental data, thereby increasing confidence that the code can reliably be used to predict the performance of other machines as well.

References

- [1] Davis, R. L., Hobbs, D. E., and Weingold, H. D., "Prediction of Compressor Cascade Performance Using a Navier-Stokes Technique," ASME Paper 88-GT-96, June 1986.
- [2] Chevrin, P.-A., and Vuillez, C., "Viscous Flow Computations in Turbomachine Cascades," ASME Paper 90-GT-76, June 1990.
- [3] Cambier, L., Escande, B., and Veuillot, J. P., "Computation of Internal Flows at High Reynolds Number by Numerical Solution of the Navier-Stokes Equations," *Rech. Aérop.* (English ed.), No. 1986, pp. 27-44.
- [4] Boyle, R. J., "Navier-Stokes Analysis of Turbine Blade Heat Transfer," ASME Paper 90-GT-42, June 1990.
- [5] Chima, R. V., "Explicit Multigrid Algorithm for Quasi-Three-Dimensional Viscous Flows in Turbomachinery," *J. Propulsion Power*, Vol. 3, No. 5, Sept.-Oct. 1987, pp. 397-405.
- [6] Pierzga, M. J., and Wood, J. R., "Investigation of the Three-Dimensional Flow Field Within a Transonic Fan Rotor: Experiment and Analysis," *J. Eng. Gas Turbines Power*, Vol. 107, No. 2, Apr. 1985, pp. 436-449.
- [7] Denton, J. D., "An Improved Time Marching Method For Turbomachinery Flow Calculations," ASME Paper 82-GT-239, Apr. 1982.
- [8] Dawes, W. N., "Towards Improved Throughflow Capability: The Use of 3-D Viscous Flow Solvers in a Multistage Environment," ASME Paper 90-GT-18, June 1990.
- [9] Adamczyk, J. J., Celestina, M. L., and Greitzer, E. M., "The Role of Tip Clearance in High-Speed Fan Stall," ASME Paper 91-GT-83, June 1991.
- [10] Chima, R. V., and Yokota, J. W., "Numerical Analysis of Three-Dimensional Viscous Internal Flows," *AIAA J.*, Vol. 28, No. 5, May 1990, pp. 798-806.
- [11] Fottner, L., ed., *Test Cases for Computation of Internal Flows in Aero Engine Components*, AGARD Advisory Report No. 275, AGARD, Neuilly-Sur-Seine, France, July 1990.
- [12] Strazisar, A. J., Wood, J. R., Hathaway, M. D., Suder, K. L., "Laser Anemometer Measurements in a Transonic Axial Flow Fan Rotor," NASA TP-2879, Nov. 1989.
- [13] Baldwin, B. S., and Lomax, H., "Thin-Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-257, Jan. 1978.
- [14] Jameson, A., Schmidt, W., and Turkel, E., "Numerical Solutions of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes," AIAA Paper 81-1259, June 1981.
- [15] Martinelli, L., and Jameson, A., "Validation of a Multigrid Method for the Reynolds Averaged Equations," AIAA Paper 88-0414, Jan. 1988.
- [16] Kunz, R. F., and Lakshminarayana, B., "Computation of Supersonic and Low Subsonic Cascade Flows Using an Explicit Navier-Stokes Technique and the $k - \epsilon$ Turbulence Model," in *Computational Fluid Dynamics Symposium on Aeropropulsion*, NASA CP-10045, Apr. 1990.
- [17] Jameson, A., and Baker, T. J., "Solution of the Euler Equations for Complex Configurations," AIAA Paper 83-1929, July 1983.
- [18] Steger, J. L., and Sorenson, R. L., "Automatic Mesh-Point Clustering Near a Boundary in Grid Generation with Elliptic Partial Differential Equations," *J. Comp. Phys.*, Vol. 33, No. 3, Dec. 1979, pp. 405-410.

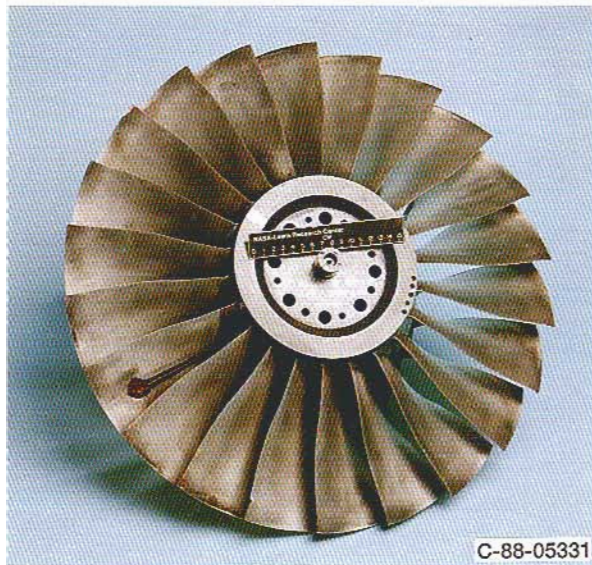


Figure 1.—Rotor 67 test hardware.

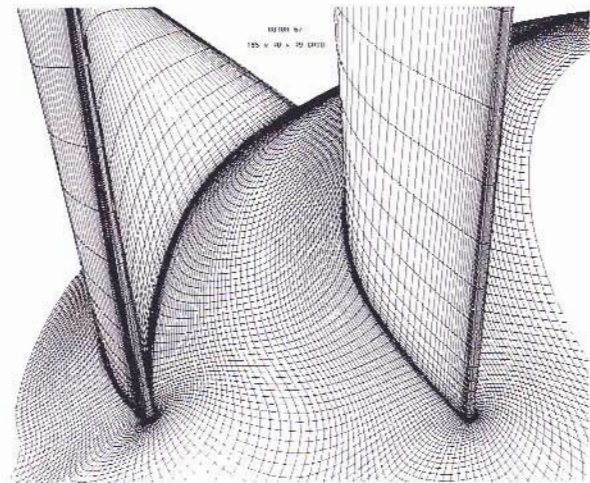


Figure 3.—Grid leading edge enlargement.

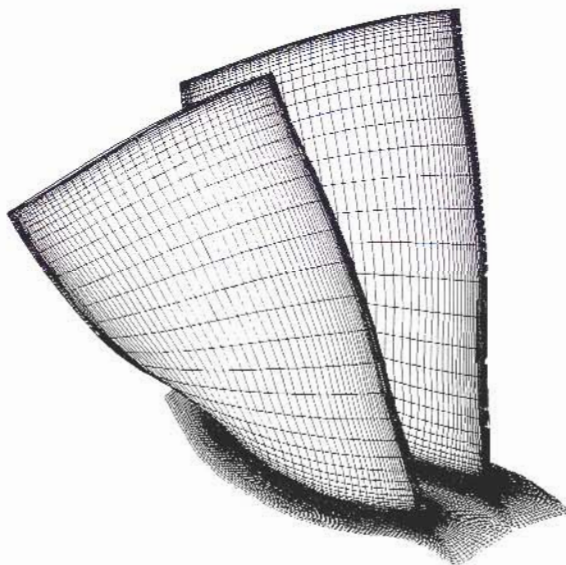


Figure 2.—Computational grid. 185 C \times 40 tangential \times 49 spanwise points.

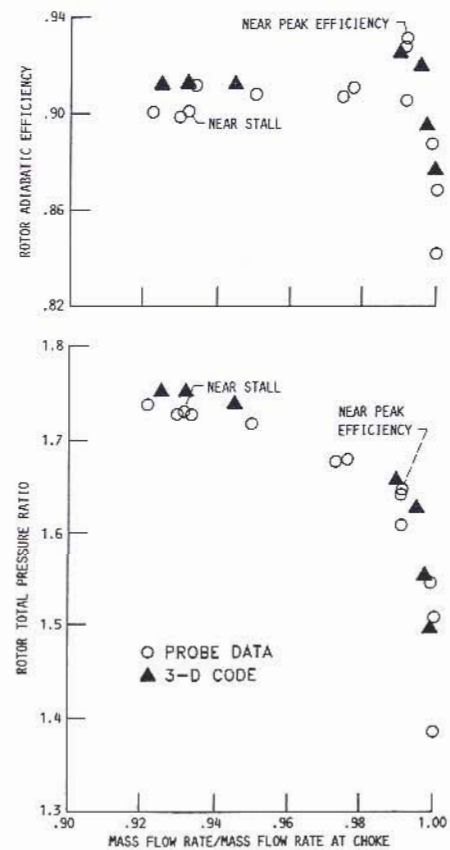


Figure 4.—Comparison of computed and measured adiabatic efficiency and total pressure ratio characteristic at 100 percent speed. Points labeled "near stall" and "near peak efficiency" are discussed in detail in the text.

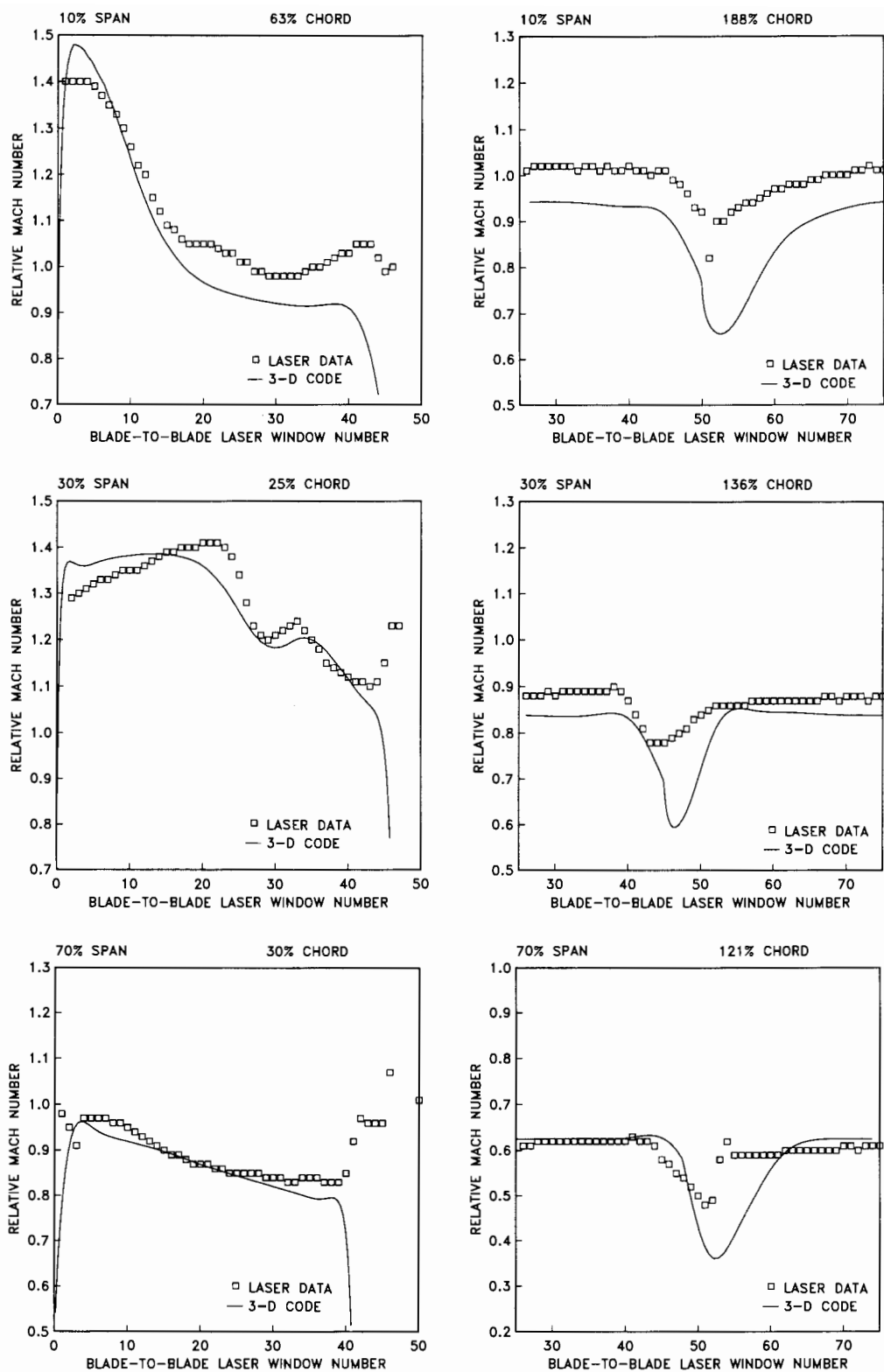


Figure 6.—Comparison of computed and measured blade-to-blade distributions of relative Mach number, near peak efficiency.

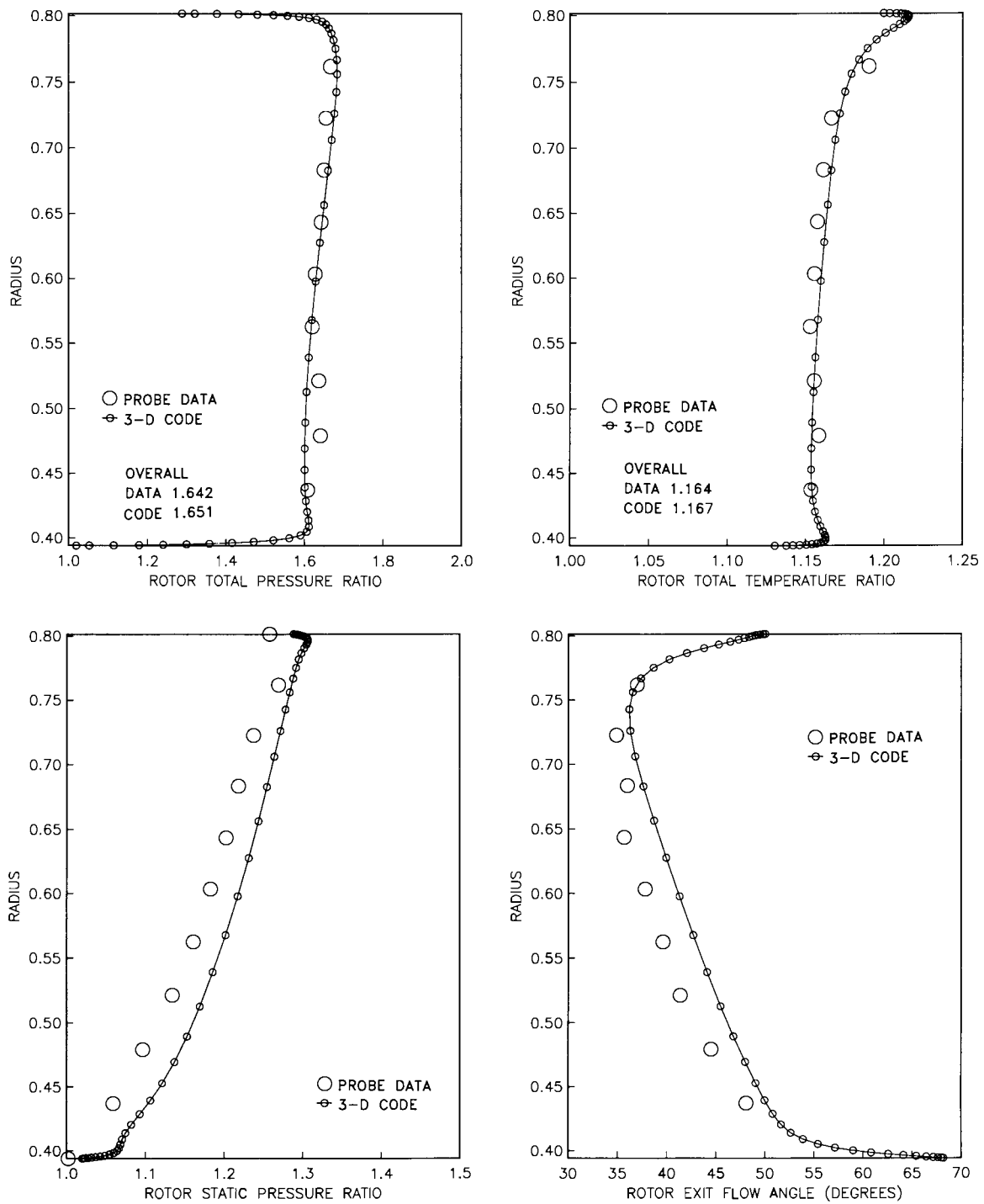
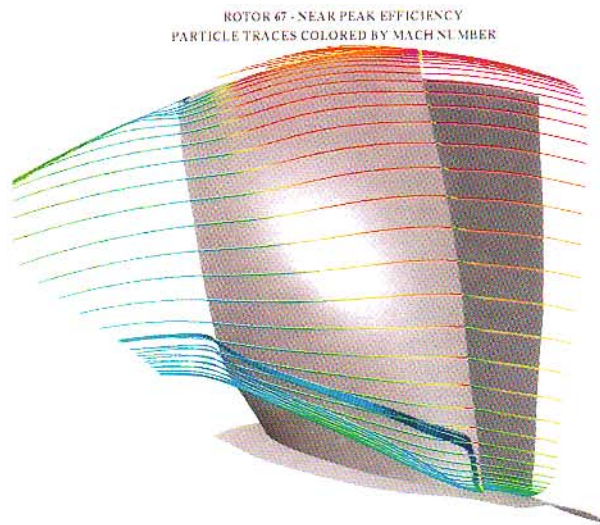
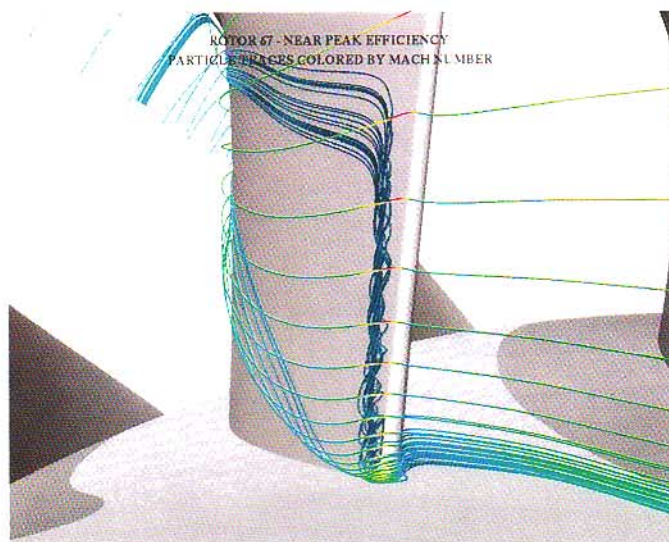


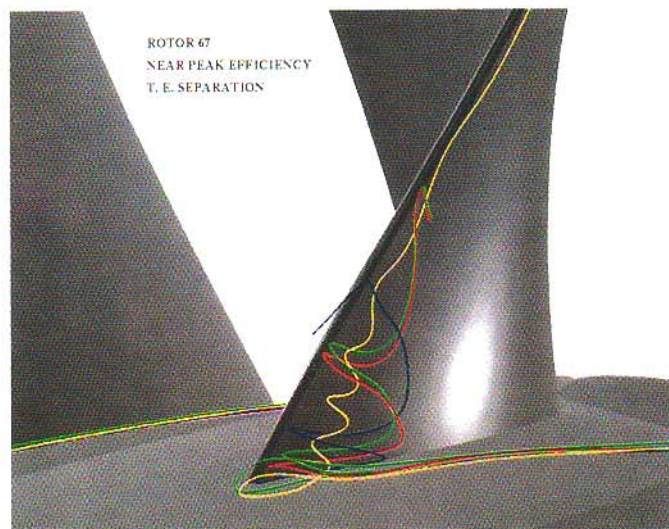
Figure 7.—Comparison of computed and measured spanwise distributions of total pressure, total temperature, static pressure, and flow angle downstream of the rotor, near peak efficiency.



(a) Suction surface overview.



(b) Leading edge enlargement, looking downstream.



(c) Trailing edge enlargement, looking upstream.

Figure 8.—Computed particle traces, near peak efficiency.

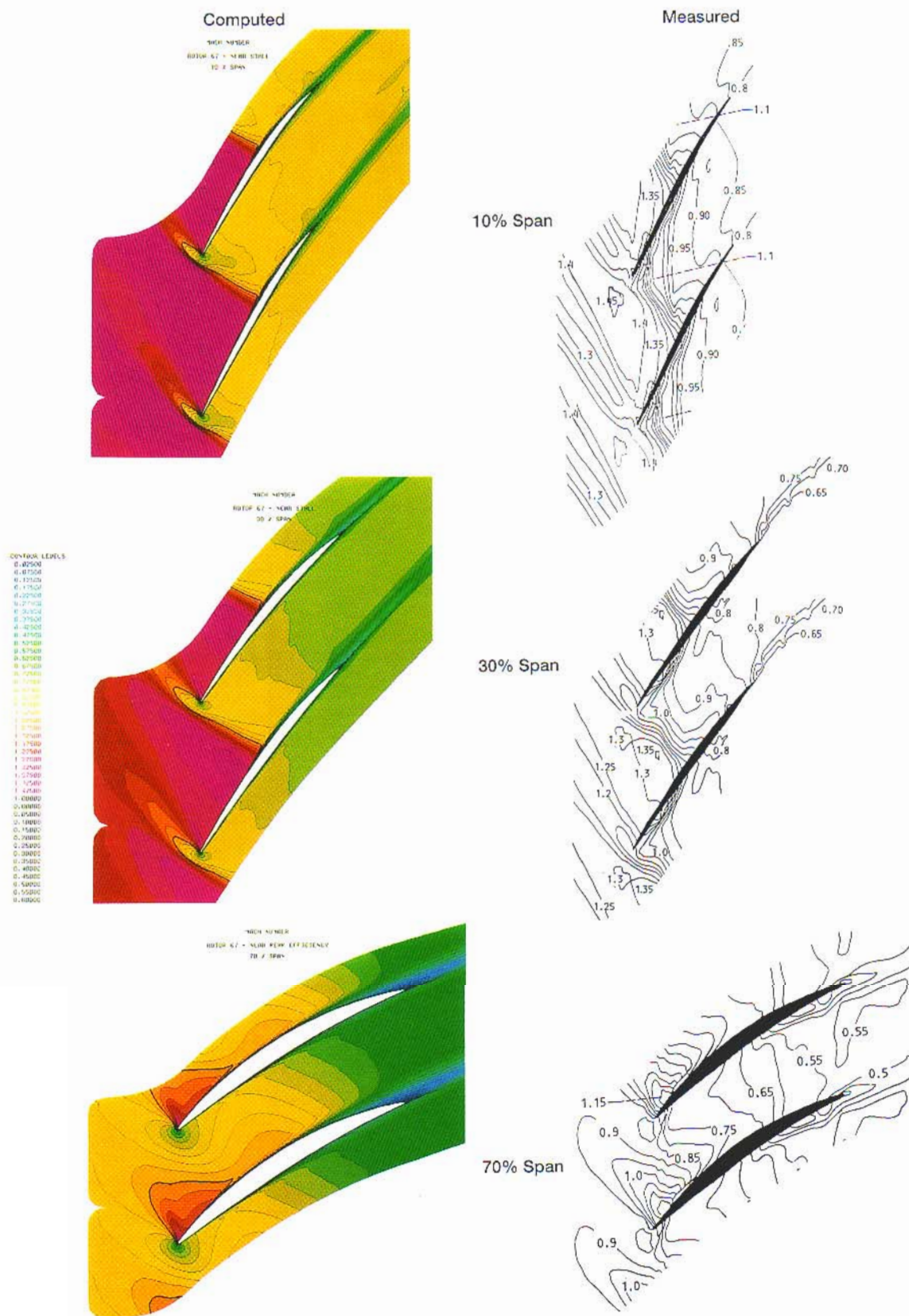


Figure 9.—Contours of relative Mach number, near stall. In computed contours heavy line is $M = 1.0$, and contour increment is 0.05.

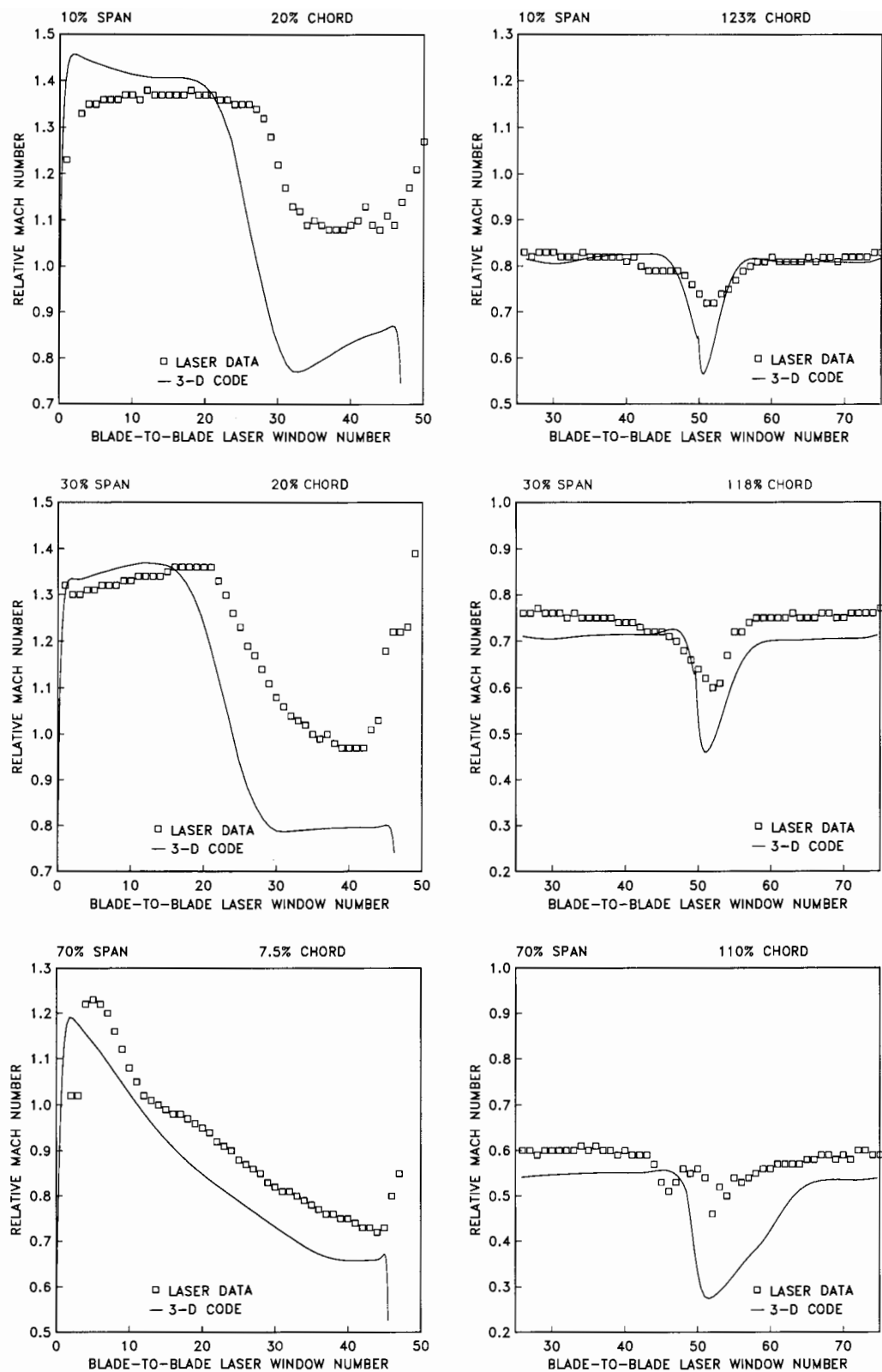


Figure 10.—Comparison of computed and measured blade-to-blade distributions of relative Mach number, near stall.

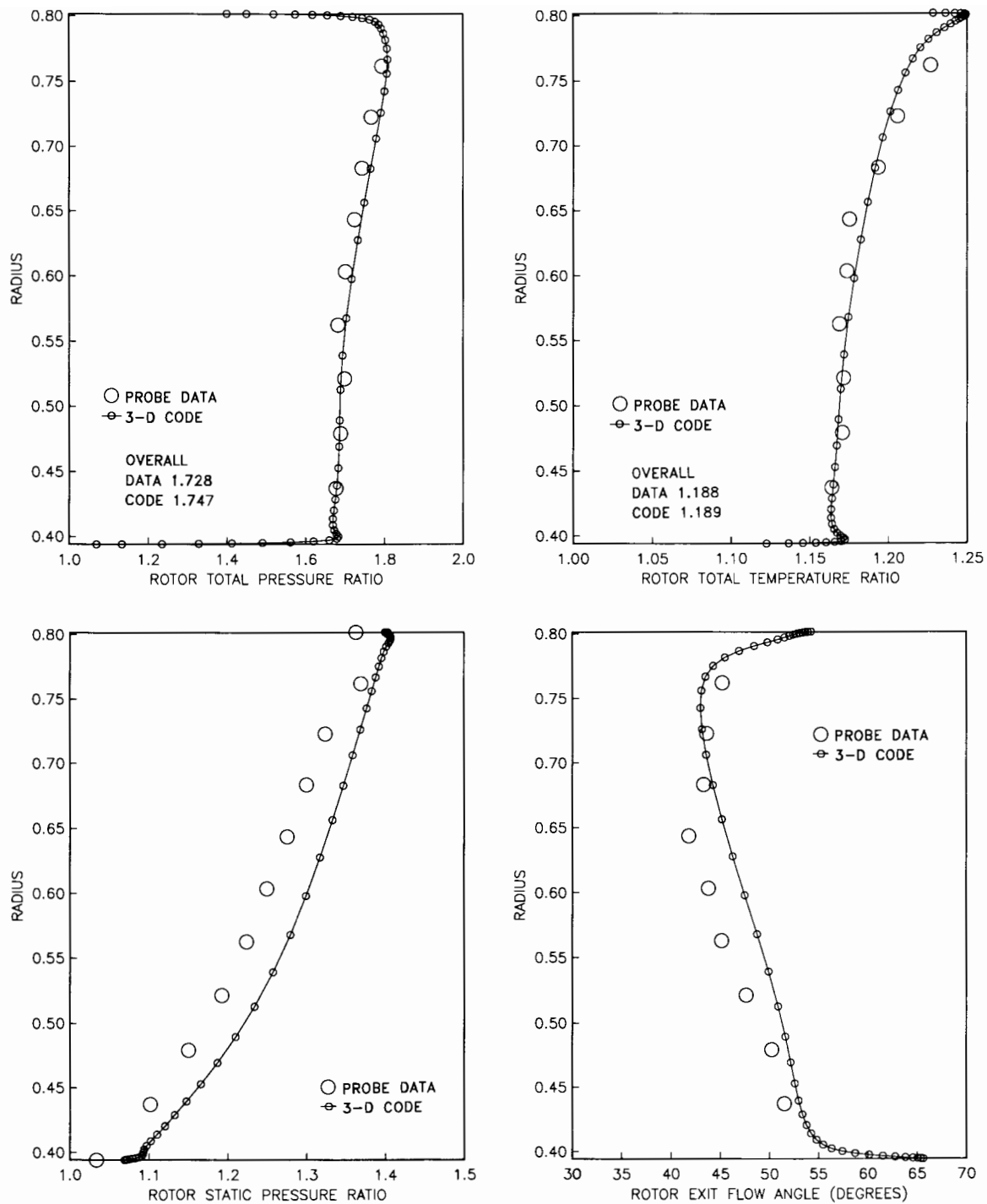


Figure 11.—Comparison of computed and measured spanwise distributions of total pressure, total temperature, static pressure, and flow angle downstream of the rotor, near stall.

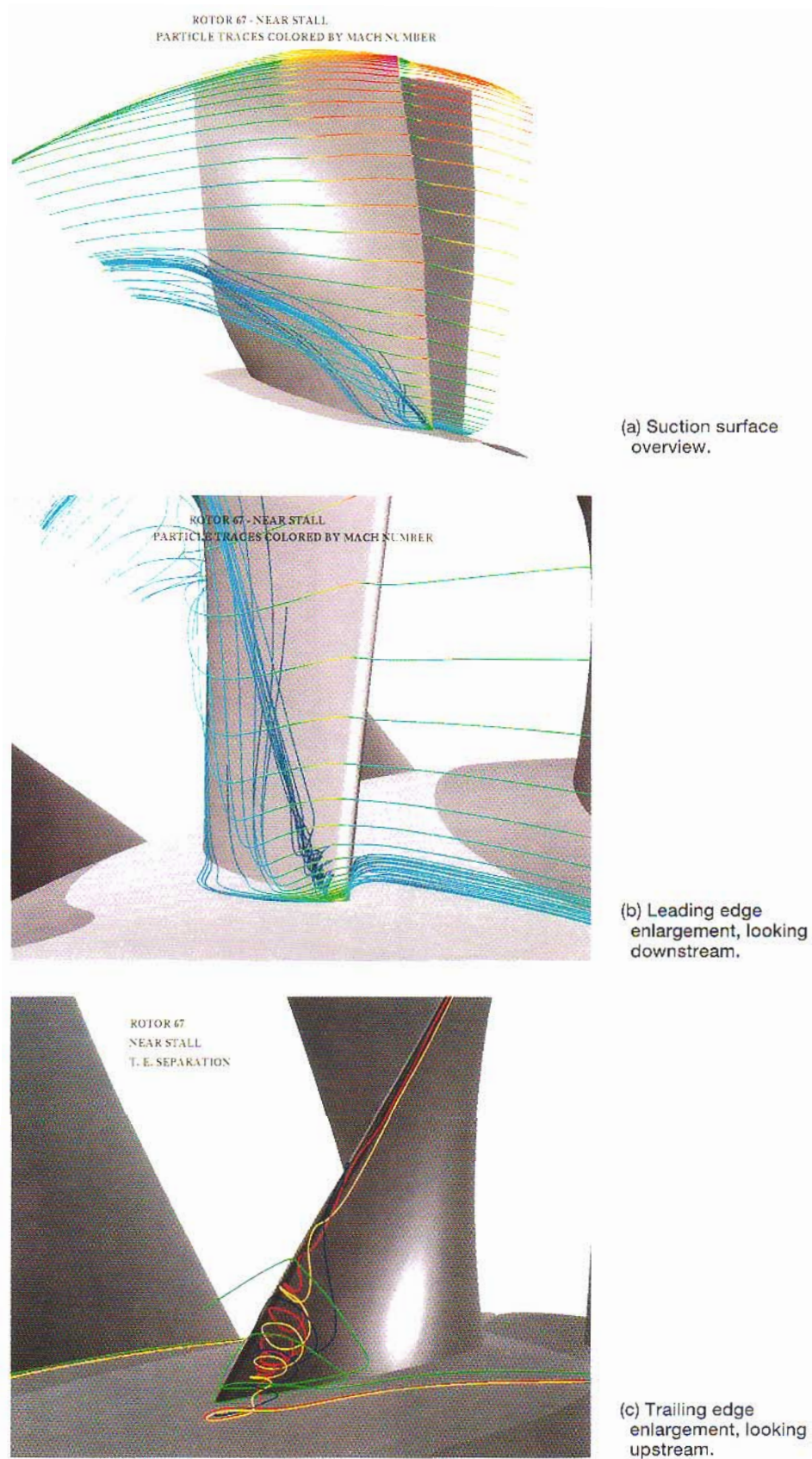


Figure 12.—Computed particle traces, near stall.

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-0188	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.				
1. AGENCY USE ONLY (Leave blank)		2. REPORT DATE		3. REPORT TYPE AND DATES COVERED Technical Memorandum
4. TITLE AND SUBTITLE Viscous Three-Dimensional Calculations of Transonic Fan Performance			5. FUNDING NUMBERS WU - 505 - 62 - 20	
6. AUTHOR(S) Rodrick V. Chima				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135 - 3191			8. PERFORMING ORGANIZATION REPORT NUMBER E - 6088	
9. SPONSORING/MONITORING AGENCY NAMES(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, D.C. 20546 - 0001			10. SPONSORING/MONITORING AGENCY REPORT NUMBER NASA TM - 103800	
11. SUPPLEMENTARY NOTES Prepared for the 77th Symposium of the Propulsion and Energetics Panel entitled CFD Techniques for Propulsion Applications sponsored by AGARD, San Antonio, Texas, May 27 - 31, 1991. Responsible person, Rodrick V. Chima, (216) 433 - 5919.				
12a. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified - Unlimited Subject Category 01			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words) A three-dimensional flow analysis code has been used to compute the design speed operating line of a transonic fan rotor, and the results have been compared with experimental data. The code is an explicit finite difference code with an algebraic turbulence model. It is described in detail in the paper. The transonic fan, designated rotor 67, has been tested experimentally at NASA Lewis Research Center with conventional aerodynamic probes and with user anemometry and has been included as one of the AGARD test cases for the computation of internal flows. The experimental data are described briefly. Maps of total pressure ratio and adiabatic efficiency versus mass flow have been computed and are compared with the experimental maps, with excellent agreement between the two. Detailed comparisons between calculations and experiment are made at two operating points, one near peak efficiency and the other near stall. Blade-to-blade contour plots are used to show the shock structure. Comparisons of circumferentially integrated flow quantities downstream of the rotor show spanwise distributions of several aerodynamic parameters. Calculated Mach number distributions are compared with laser anemometer data within the blade row and the wake to quantify the accuracy of the calculations. Finally, particle traces are used to illustrate the nature of the secondary flow in this fan.				
14. SUBJECT TERMS Navier-Stokes equation; Turbomachinery; Computational fluid dynamics; Three-dimensional flow			15. NUMBER OF PAGES 20	
			16. PRICE CODE A03	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT	